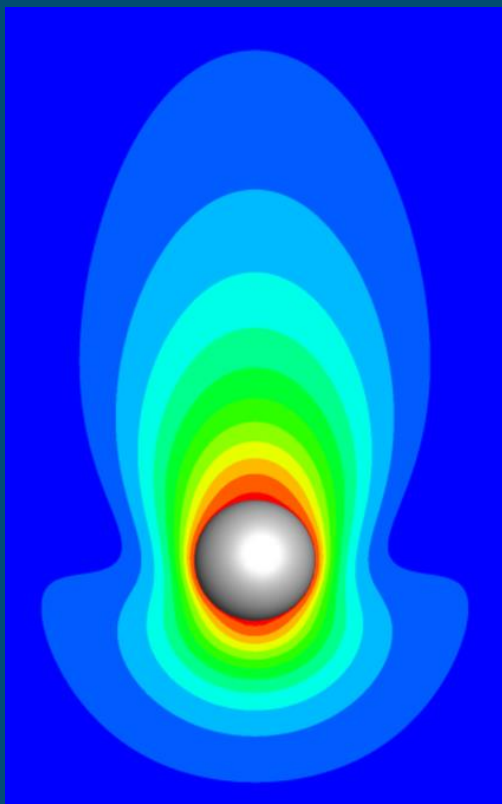# TUDUBLIN

CFD Research Group Guide Series

Verification of Laminar Pipe Flow CFD Test Cases Guide

Author: Gerald Gallagher

Rev. 5

This guide contains information on how to verify laminar pipe flow CFD test cases, and includes diagrams, equations and website links as necessary to compliment the explanations. In Fig. 1 below, the developing velocity profile of a fluid entering a pipe is depicted which should help students visualise the velocity profile. This guide will likely be updated in the future to add further information based on student feedback, but there should be more than enough information available currently to act as a starting point for robust validation of laminar pipe flow simulations.
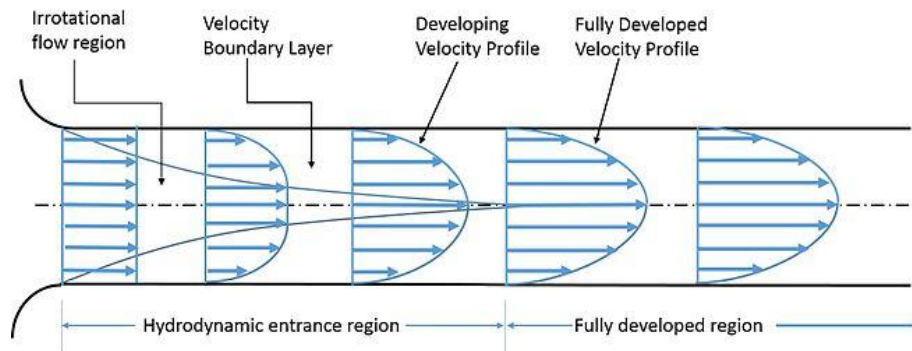
Figure 1: The developing velocity profile of a fluid entering a pipe

<u>**Reynolds Number**</u>

Used to determine whether the flow is laminar or turbulent (less than 2000 is laminar, over 4000 is turbulent).

$$Re = \frac{\rho u_{avg} D}{\mu} = \frac{u_{avg} D}{\nu}$$

ρ = density (kg/m^3)

u_{avg} = average velocity (m/s)

D = characteristic dimension (m)

*In the case of a circular pipe, the characteristic length is the diameter. For non-circular cross sections, use hydraulic diameter

μ = absolute or dynamic viscosity (kg/ms)

ν = kinematic viscosity (m²/s)

<u>Hydrodynamic Entrance length</u>

If a constant velocity is applied across the diameter at the inlet to your pipe (see the diagram above), a hydrodynamic entrance length is required to calculate the distance from the inlet to the point in the pipe that the flow is considered to be fully developed (important for comparing against analytical velocity equation). The hydrodynamic entrance length for laminar flow in a pipe can be calculated using the following equation:

$$L_{laminar} = 0.0575 \, Re \, D$$

For a pipe with a non-circular cross-section, you should replace D with D_h where:

$$D_h = \frac{4A}{P}$$

A = cross-sectional area of the flow (m²)

P = wetted perimeter of the cross-section (m)

D_h for a square duct:	D_h for a rectangular duct (fully filled):
$$D_H = \frac{4a^2}{4a} = a$$ a = length of duct side (m)	$$D_H = \frac{4ab}{2(a+b)} = \frac{2ab}{a+b}$$ a = height of duct (m) b = width of duct (m)

Poiseuille Equation

The Poiseuille equation gives the pressure drop for fully developed incompressible laminar flow in a long cylindrical pipe of constant cross section for a Newtonian fluid.

$$\Delta p = \frac{8\mu L Q}{\pi R^4} = \frac{8\pi\mu L Q}{A^2}$$

Δp = pressure drop along the length of the pipe (Pa)
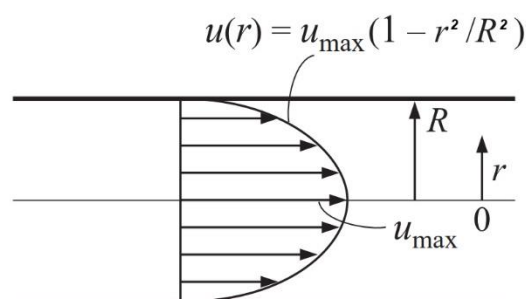
L = length of pipe section (m)

Q = volume flow rate (m³/s); $Q = VA$ so if V (in this case u_{avg}) and A are known, Q can be obtained

R = pipe radius (m)

A = cross-sectional area of the pipe (m²)

Flow Profile for Fully Developed Laminar Flow in a Pipe

Fig.2 shows depicts a fully developed velocity profile with an expression for obtaining the velocity profile along the diameter of the pipe.

Figure 2: Mathematical description of fully developed velocity profile

$$u(r) = u_{max}\left[1 - \left(\frac{r}{R}\right)^2\right]$$

$u(r)$ = velocity at any point r along the diameter of the pipe (note that r = 0 at the centreline so moving upwards from the centreline will give a positive r value and moving downwards from the centreline will give a negative r value)

$$u_{max} = 2u_{avg} \therefore u_{avg} = \frac{u_{max}}{2}$$

r = extending radially from the centreline, chosen point along the radius of the pipe

R = fixed distance from centreline to pipe wall

The derivation of this formula (based on a polar coordinate system) is shown in the video https://youtu.be/LfBDyeaaAoA. Also, note that the above equation for $u(r)$ can be applied to 2D simulations, however u_{avg} changes as follows:

$$u_{max,2D} = \frac{2}{3}u_{avg,2D} \therefore u_{avg,2D} = \frac{3}{2}u_{max,2D}$$

Guidelines for Approaching the Simulations

1. Look at different sources (YouTube videos can be useful) to learn how to create your pipe geometry (DesignModeler or SpaceClaim is recommended for simple geometries). Here are some good YouTube videos with some useful information in relation to Ansys Fluent:

    a. https://youtu.be/FjKMjnhKThY - (3D) Ansys Fluent laminar pipe flow tutorial

    b. https://youtu.be/yIa18xoveoc - (3D) demonstrates how to add a parabolic velocity profile to the inlet without using a User Defined Function (UDF) i.e. uses an expression

    c. https://youtu.be/LF06grv299A - (2D) demonstrates how to add a parabolic velocity profile to the inlet without using a User Defined Function (UDF) i.e. uses an expression

      d.   https://youtu.be/XKmcISqR9Q4 - further information on expressions

2. Once you are able to set up a basic simulation with a geometry and inlet, outlet and wall boundary conditions, you can proceed to apply the previously defined equations and ideally do all of the following:

    a.   Specify a max velocity (u_{max}) in m/s and calculate u_{avg}

    b.   Calculate a Reynolds number based on the chosen fluid properties and the average velocity in the pipe

    c.   Calculate a hydrodynamic entrance length

    d.   Modify the length of your pipe geometry to match the hydrodynamic entrance length

    e.   Set an inlet velocity boundary condition using u_{avg} (note that you need to set the velocity component parallel to the pipe walls to u_{avg} with the other components being zero. In some videos the author sets a velocity magnitude; if this option is used, under "Velocity Specification Method", "Magnitude, Normal to Boundary" must be selected)

    f.   The outlet condition should be a simple pressure condition, and the walls should use no-slip velocity conditions

    g.   After running the simulation, you should be able to extract plots of the velocity profile at the end of the pipe and the pressure drop along the pipe (you should be able to do this in Ansys Fluent after reading the documentation, completing tutorials and following various videos)

    h.   The velocity profile should be compared against the values obtained for $u(r)$ using the equation defined previously

    i.   If a fully developed velocity profile was specified at the inlet instead of u_{avg} (look at the previous videos for help with this), the Poiseuille equation defined previously can be used to find Δp and compare it against the pressure drop

obtained from the simulation. In this case, the hydrodynamic length is not important (although the ratio of the pipe length to its radius should be greater than 1/48 for the equation to be valid)

For any simulation, mesh convergence studies and percentage errors between the calculations and the simulation results should be explicitly shown to prove that your simulations are working. For information on this, ask your supervisor!